Thermal Induced Stresses on a PCB

Problem Description

This tutorial describes the steps to perform a thermal-structural simulation of a printed circuit board using ANSYS SpaceClaim and ANSYS Mechanical.

The tutorial shows you how to 1.) prepare a geometry in ANSYS SpaceClaim so that it works in cooperation with the Trace Mapping feature of ANSYS Mechanical and 2.) demonstrates the use of the Trace Import feature by examining the warpage (deformation) of a simply supported printed circuit board (PCB) as a result of uniform thermal loading.

Features Demonstrated

- Engineering Data/Materials
- Static Structural Analysis
- Electronic Computer-Aided Design (ECAD)
- Trace Mapping

Procedure

- 1. Set Up the Analysis.
 - a. Open ANSYS Workbench.
 - b. Create a Static Structural analysis and name it "ECAD." You may create any name as desired.

Project	Project Schematic						
		1			_		
	•		A				
	1	2	Static Structural				
	2	٢	Engineering Data	\checkmark	4		
	3	\bigcirc	Geometry	?	4		
	4	6	Model	ę	4		
	5		Setup	?	4		
	6	6	Solution	7	4		
	7	۲	Results	7	4		
			ECAD				

2. Select Materials in Engineering Data.

a. Open the Engineering Data Workspace: right-click on the Engineering Data cell and select Edit.

Project	Sch	nemat	ic			
					_	
	▼		A			
	1	7	Static Structural			
	2		Engineering Data	\checkmark		Edit
	3	\bigcirc	Geometry	?		
	4	۲	Model	7	•	Duplicate
	5	٢	Setup	?		Transfer Data From New
	6		Solution	7	7	Update
	7	۲	Results	?		Update Upstream Components
			ECAD		\$	Refresh
					ab	Rename
						Properties
						Quick Help
						Add Note

b. Select and delete Structural Steel (right-click>Delete), the default material.



- c. Select the Engineering Data Sources button on the toolbar.
- d. From the **General Material** library, add **FR4** (Dialectic Material) and **Copper Alloy** using the plus sign button in the **Add** column. A book icon displays in the **Add** column when you select the material.

1	Project 🍕	A2	2:Engir	eering D	ata 🗙				
T Filt	er Engineering Data 🧾 Engineeri	ing Da	ata Sou	irces					
Engineer	ring Data Sources					<u>~</u> ⋣ X			
	A		В	С		D			
1	Data Source			Location	۱	Description			
2	🚖 Favorites				Quick access list and	default items			
3	🎒 General Materials				General use material	samples for use in various analyses.			
4	General Non-linear Materials	;			General use material	samples for use in non-linear analyses.			
5	Explicit Materials				Material samples for	use in an explicit analysis.			
6	Hyperelastic Materials				Material stress-strain	data samples for curve fitting.			
7	Magnetic B-H Curves				B-H Curve samples sp	pecific for use in a magnetic analysis.			
8	🎒 Thermal Materials				Material samples spec	ific for use in a thermal analysis.			
9	Fluid Materials				Material samples spe	tific for use in a fluid analysis.			
10	📁 Composite Materials				Material samples spe	ific for composite structures.			
*	Click here to add a new library								
Outline o	of General Materials					, ∓ Χ			
	А	в	С		D	E			
1	Contents of General 🕒	A	dd		Source	Description			
2	= Material								
3	2 📎 Air	÷		🚆 Ge	neral_Materials.xml	General properties for air.			
4	Numinum Alloy	÷		🚆 Ge	neral_Materials.xml	General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.			
5	Soncrete	÷		🚆 Ge	neral_Materials.xml				
6	📎 Copper Alloy	+	۲	🚆 Ge	neral_Materials.xml				
7	📎 FR-4	4	۲	🔮 Ge	neral_Materials.×ml	Sample FR-4 material, data is averaged from various sources and meant for illustrative purposes. It is assumed that the material × direction is the length-wise (LW), or warp yan direction, while the material y direction is the cross-wise (CW), or fill yarn direction.			
8	📎 Gray Cast Iron	+		🚆 Ge	neral_Materials.xml				
9	📎 Magnesium Alloy	÷		😤 Ge	neral_Materials.xml				
10	📎 Polyethylene	÷		🚆 Ge	neral_Materials.xml				
11	🦠 Silicon Anisotropic	÷		🚆 Ge	neral_Materials.xml				
12	📎 Stainless Steel	4		🚆 Ge	neral_Materials.xml	I_Materials.xml			
13	🦠 Structural Steel	÷		Ge	neral_Materials.xml	Materials.xml Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1			
14	📎 Titanium Alloy	÷		宁 Ge	neral_Materials.×ml				

e. Select the **Engineering Data Sources** button. The new materials display in the **Outline of Schematic Pane** and will now be available in Mechanical.

Outline	of Schematic A2: Engineering Data			
	А	в	С	D
1	Contents of Engineering Data 🗦	9	8	Source
2	Material			
3	📎 Copper Alloy	•		General_Materials.xml
4	🦠 FR-4	•		General_Materials.xml
*	Click here to add a new material			

f. Return to the Workbench Project page.

3. Define Geometry.

a. Right click on the **Geometry** cell and select **Edit Geometry in SpaceClaim**.

Sch	nemat	ic			
•		A			
1	~	Static Structural			
2	0	Engineering Data	× .		
3	\bigcirc	Geometry	? 🖌	6	New DesignModeler Geometry
4	۲	Model	? 🖌	57	New SpaceClaim Geometry
5		Setup	?		
6	6	Solution	?		Import debined y
7	6	Results	?		Duplicate
		ECAD			Transfer Data From New Transfer Data To New
				7	Update
					Update Upstream Components
				\$	Refresh
					Reset
				ab	Rename
					Properties
					Quick Help
					Add Note

The SpaceClaim application opens.

- b. In SpaceClaim, select File>Open.
- c. From the **Open** dialog box, select the **Options** button and verify that the Layer Topology option is selected under **File Options>ECAD**. Click **Ok**.

a constant				SZ PClaim
Upen		Succession Outlines		
Libraries Documer	ts 🕨	spacecialm Options		£ ×
Organize 🔻 New folder		Popular	STAD file options	lit Body
Image: Second conditions Image: Second conditions Image: Second conditions	nents library 2 locations	 Detailing Appearance Selection Snap Units Sheet Metal Navigation Advanced File Options General ACIS AMF Auto CAD CATIA ECAD 	Import mode Layout Geometry Layer Topology Create solderballs as Bounding Box Cylinders	lit pject
File name:		IGES		- × - 2
		JT Open		
		ОВЈ		
Lock base point		Parasolid		
🔀 Sketch 🛞		PDF		
Snap to grid Snap to angle Create layout curves Properties		Pro/ENGINEER Rhino SketchUp 🗸		
	z × ×		ОК	Cancel
Properties Appearance	∧ Design1×			
Ready				●▲ -

d. From the **Open** dialog box, import the geometry file provided: file name ECAD_Tutorial.tgz.This file is available on the ANSYS Customer Portal.

🔊 Open ເ⊖⊖ ⊖ マ 📕 « Tu	utorial 🕨 ECAD Tutorial 🕨	✓ 4y Search ECAD Tutorial
Organize - Ne	ew folder	III 🔹 🖬 🔞
☆ Favorites ■ Desktop ● Downloads ● Recent Places	Documents library ECAD Tutorial Name	Arrange by: Folder ▼
 Libraries Documents Music Pictures Videos 	E	
	Options File <u>n</u> ame: ECAD_Tutorial.tgz	✓ Other ECAD (*.tgz;*.xml;*.cvg;*. ▼ Qpen ▼ Cancel

e. Uncheck the **Components (STP)** object in the tree. They will not be needed in Mechanical. Select and view the geometries as desired.



f. Select the STP object and then set the Share Topology property to Share.

Structure ¹
🛛 📝 Design2*
⊿ 🔽 🍣 STP
🔽 🗊 top
V 🗊 Dielectric_1
🗹 🗊 int1
Dielectric_2
V 🔰 int2
Dielectric_3
Dottom
Components (STP)
Structure Layers Selection Groups Views
Properties +
▲ Analysis
Share Topology Share
▷ File
⊿ Material
Material Name Unknown Material
▷ Name
Sheet Metal

- g. Save the file as ECAD_Tutorial_File.scdoc.
- h. Return to the Workbench Project page.

4. Import the Geometry into Mechanical.

a. Place an **External Data** system into the project and drag it in front of the Static Structural system.

Project Schematic						
▼ A		-		В		
1 🔁 External Data .		1	777	Static Structural		
2 🍓 Setup	? 🖌	2	٢	Engineering Data	 _ 	
External Data		3	\bigcirc	Geometry	? 🖌	
		4	6	Model	2 🖌	
		5	٢	Setup	? 🖌	
		6	6	Solution	? 🖌	
		7	1	Results	? 🖌	
				ECAD		

- b. Right-click on the Setup cell and select Edit.
- c. Select the button in the Location column, browse to the ECAD_Tutorial.tgz file, and open it.

1)	🗋 🚰 🛃 🔲 Project 🔮 A:External Data 🗙										
Outline o	Outline of Schematic A2 : 🗾 👻 📮 🗙										
	А	В		с		D		E			
1	Data Source 🛛 💌	Locatio	on	Identifier	•	Master	•	Desc	ription	•	
2	Click here to add a file										
				Browse							
				Browse fr	om F	Repository					

d. Select the row for the file in the **Outline** to display the properties. As needed, specify **ODB++TGZ** for the **Format Type**. Note the default Identifier, **File1**.

1	🗋 📴 🛃 🔲 Project 🍓 A:External Data 🗙						
Outline o	Outline of Schematic A2 :						
		A			в	С	
1		Data Source		-	Location	Identifier 💌	
2	C:\ECAD_Tutorial.	tgz				File1	
3	Click here to add a	file					
Propertie	roperties of File - C:\ECAD_Tutorial.tgz						
	А	В	С				
1	Property	Value	Unit				
2	Definition						
3	Format Type	ODB++ TGZ 💌					
4	😑 Rigid Transformati	on					
5	Origin X	0	m 💌	1			
6	Origin Y	0	m 💌	1			
7	Origin Z	0	m 💌	1			
8	Theta XY	0	radian 💌	1			
9	Theta YZ	0	radian 👱	1			
10	Theta ZX	0	radian 💌	1			

- e. Return to the Workbench Project page.
- f. Select the **Update Project** button.
- g. Drag and drop the **Setup** cell onto the **Model** cell of the Static Structural system to create a link.



- h. Right-click on the **Geometry** cell and browse (**Import Geometry**>**Browse**) to the geometry file you saved in SpaceClaim (ECAD_Tutorial_File.scdoc) and open it.
- i. Right-click on the **Model** cell and select **Edit** to open the files in Mechanical.

oject Schematic		
1 External Data 1 27 Static Structural		
2 🖗 Setup 🗸 🚽 2 🥥 Engineering Data 🗸		
External Data 3 @ Geometry ?		
•4 Wodel		Edit
5 🌒 Setup 🖀		
6 🎧 Solution 🔗	W	Edit in Read-Only Mode
7 🥪 Results 😭	• •	Duplicate
ECAD		Transfer Data From New
		Transfer Data To New
	7	Update
		Update Upstream Components
		Clear Generated Data
	4	Refresh
		Reset
	ab	Rename
		Properties
		Quick Help
		Add Note

- 5. Specifying Materials. In Mechanical you will note that the Geometry object is underdefined.
 - a. Open the **STP** object and select all of the child objects. Select **FR-4** from the drop-down list of the **Assignment** property.

Outline		Д.					
Filter Name							
] 🕼 🖉 từ 🖽 🛃							
Project							
Email Geometry							
	TP\Dielectric_1						
	STP\int1						
	STP\Dielectric_2						
	STP \int2						
	STP/Dielectric_3						
? 🛡 : Elimont	ted Trace (Setup 1)						
E Coordinate	Systems						
Connections	3						
💞 Mesh							
📄 🦙 🦲 Static Stru	uctural (B5)						
Analys	sis Settings						
⊡?® Solut	ion (B6)						
······· / 11 :	solution Information						
Details of "Multiple Selecti	ņ						
Graphics Properties							
 Definition 							
Suppressed	No						
Stiffness Behavior	Flexible						
Coordinate System	Default Coordinate System						
Reference Temperature	By Environment						
Behavior	None						
Material							
Assignment		C R	Now Material				
Nonlinear Effects	®	June of the second seco					
Thermal Strain Effects	Thermal Strain Effects Yes 🎽						
Bounding Box		۲	FR-4				
Properties			> Copper Alloy				
Statistics		Ť					

- b. Open the Imported Trace folder and select the Imported Trace object.
- c. Right-click in the Geometry window and select the option Select All.



d. Click the **Geometry** property in the Details view and click **Apply**. Seven bodies are specified for the **Geometry** property.



e. Select File1 (from the External Data system) for the External Data Identifier property.

0	utline		Ļ						
] F	Filter: Name 🗸			In	mp	orted Irace			
]	😰 🔄 🔶 🗄 🔕					Imported Tra	ice in the second s		
	Project Model (B4) Model (B4) Model (B4) Profile Geometry Profile Connections Model Connections Mode	ted Trace (Setup 1) Imported Trace Systems a uctural (B5) sis Settings ion (B6) Solution Information							
D	etails of "Imported Trace	"	ф.	Ge	еоп	netry Print	Preview A Report	Preview/	
	Scope			Data	a Vi	ew			
	Scoping Method	Geometry Selection							
	Geometry	7 Bodies		10	nμ	ported I	race		
Ξ	Definition								
	Туре	Imported Trace			_	Layer	Thickness (mm)	Trace Material	Active
	Suppressed	No			1	top	0.0355600007		
	External Data Identifier	File1	•		2	Dielectric_1	0.1193799973		<u> </u>
	Graphics Controls				3	int1	0.0355600007		
	Component	Average			4	Dielectric_2	1.1175999641		<u> </u>
ĺ	Display Source Points	Off			5	int2	0.0355600007		
÷	Settings				6	Dielectric_3	0.1193/99973		벌
Ð	Legend Controls				7	bottom	0.0355600007		
+	Named Selection Creati	on							

- f. In the Imported Trace Data View, select the Trace Material field and specify Copper Alloy.
- g. Once specified, right-click on the field again and select **Copy**. Select the remaining Trace Material fields using the Shift key, right-click again, and select paste.

All of the remaining cells populate with the **Copper Alloy** material.

Data View	Data View	Data View			
Imported Trace	Imported Trace	Imported Trace			
Layer Thickness (mm) Trace Material Active	Layer Thickness (mm) Trace Material Active	Layer Thickness (mm) Trace Material Active			
1 top 0.0355600007	1 top 0.0355600007 Copper Alloy	1 top 0.0355600007 Copper Alloy			
3 int1 0.0355600007 FR-4	3 int1 0.0355600007	3 int1 0.0355600007 Copper Alloy			
4 Dielectric_2 1.1175999641	4 Dielectric_2 1.1175999641	4 Dielectric_2 1.1175999641 Copper Alloy			
5 int2 0.0355600007	5 int2 0.0355600007	5 int2 0.0355600007 Copper Alloy 🔽			
6 Dielectric_3 0.1193799973	6 Dielectric_3 0.1193799973	6 Dielectric_3 0.1193799973 Copper Alloy			
7 bottom 0.0355600007	7 bottom 0.0355600007	7 bottom 0.0355600007 Copper Alloy			
	В Сору				
	Paste				

6. **Define Trace Properties**.

a. Specify the X-Discretization and Y-Discretization properties as 400.

D	etails of "Imported Trace	e" #				
Ξ	Scope					
	Scoping Method	Geometry Selection				
	Geometry	7 Bodies				
Ξ	Definition					
	Туре	Imported Trace				
	Suppressed	No				
	External Data Identifier	File1				
	Graphics Controls					
	Component	Average				
	Display Source Points	Off				
Ŧ	Settings					
Ŧ	Legend Controls					
Ŧ	Named Selection Creation					
	Material					
	Modeling	Averaged				
	Discretization					
	X-Discretization	400				
	V-Discretization	400				

b. Set the **Display Source Points** property to **On** to view the alignment of the source points provided by the trace layout files. Rotate the model and zoom in to view the points. Once you have finished, return the **Display Source Points** property to the **Off** setting.



Note

If you ever encounter misaligned source points in a simulation, you can use the Rigid Transformation controls in the External Data system to align the source mesh with the target.

7. Define Mesh Properties.

- a. Select the **Mesh** object.
- b. Under the **Sizing** category, specify the **Relevance Center** property as **Fine** and the **Element Size** as **0.47640** (mm). These actions refine the mesh.

Display				
Display Style	Body Color			
Defaults				
Physics Preference	Mechanical			
Relevance	0			
Shape Checking	Standard Mechanical			
Element Midside Nodes	Program Controlled			
Sizing				
Size Function	Adaptive			
Relevance Center	Fine			
Element Size	0.47640 mm			
Initial Size Seed	Active Assembly			
Smoothing	Medium			
Transition	Fast			
Span Angle Center	Coarse			
Automatic Mesh Based Defeaturing	On			
Defeaturing Tolerance	Default			
Minimum Edge Length	3.556e-002 mm			
Inflation				
Advanced				
+ Statistics				

c. Right-click on the **Imported Trace** object and select the **Import Trace** option. This mapping process will take several moments to complete. Once complete the mapping should appear as illustrated in the following image.



8. Specify Boundary Conditions.

a. Select the vertex from the bottom corner illustrated below and add a Fixed Support.



b. Apply **Displacement** boundary conditions to the bottom two corners illustrated below. Specify a 0mm displacement for the **Y Component** and **Z Component** of the first displacement and a 0mm displacement for the **Z Component** of the second displacement.



c. Apply a **Thermal Condition** load to all bodies (**Ctrl+A**) of the model and specify a temperature magnitude of 50°.



9. Specify Solver Type.

- a. Select the Analysis Settings object.
- b. Set the **Solver Type** property to **Iterative**.

Outline									
Filter: Name 🔻									
🕼 🖉 🗠 🗄 盲									
Project									
🗄 🖷 👰 Model (B4)									
🗄 ····· 🖓 Geometry									
Coordinate Systems									
Connections									
Static Structural (B5)									
Analysis Settings									
Fixed Support									
Displacement_1									
Disp	, Displacement_2								
Solution (B6)									
	bolddorffriornadorf								
Details of "Analysis Setti	ngs"	ņ							
Step Controls									
Number Of Steps	1.								
Current Step Number	1.								
Step End Time	1. s								
Auto Time Stepping	Program Controlled								
Solver Controls									
Solver Type	Program Controlled								
Weak Springs	Program Controlled								
Solver Pivot Checking	Iterative								
Large Deflection	Off								
Inertia Relief	Off								
Restart Controls	Restart Controls								
Nonlinear Controls	Nonlinear Controls								
Output Controls	Output Controls								
Analysis Data Manage	Analysis Data Management								
+ Visibility									

10. Generate Solution and Define Results.

- a. Solve the analysis. This process will take several minutes.
- b. Apply results as desired.

Total Deformation, Directional Deformation (Z Axis), and Equivalent Strain are illustrated below.







End of tutorial.